ECEN 250 Lab3 - Operational Amplifiers

Name: Brodric Young

Purposes:

- Simulate the inverting and non-inverting gain configurations of an op amp using a SPICE model
- Simulate and construct a differential amplifier using the LF347 op amp
- Make op amp DC measurements
- Extra #1: Make op amp AC measurements
- Extra #2: Simulate a Voltage Dependent Voltage Source (VDVS) model of an operational amplifier

Procedure:

<u>Part 1a - SPICE simulation of an inverting amplifier</u>
Simulate the following circuit in LTspice:

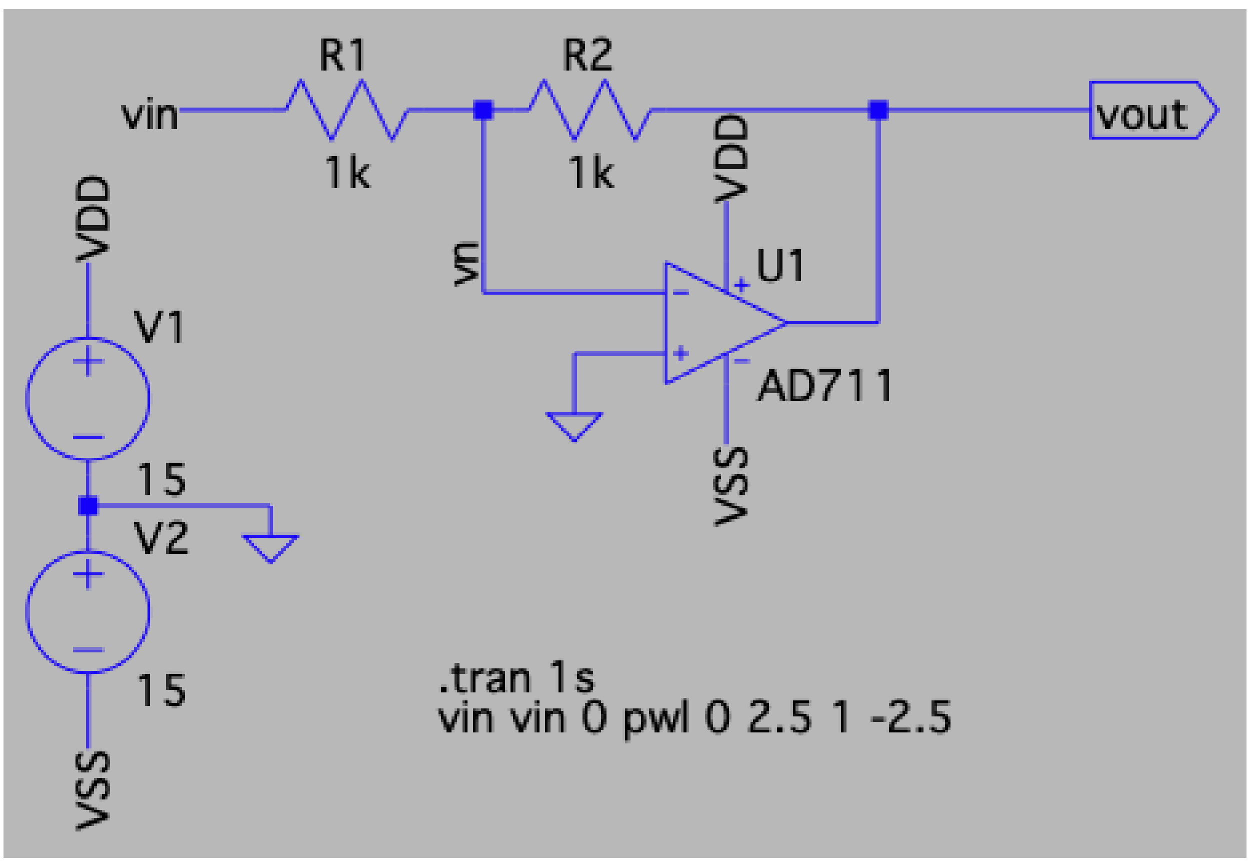


Figure 1 - A SPICE Model of an Operational Amplifier with Av = -1

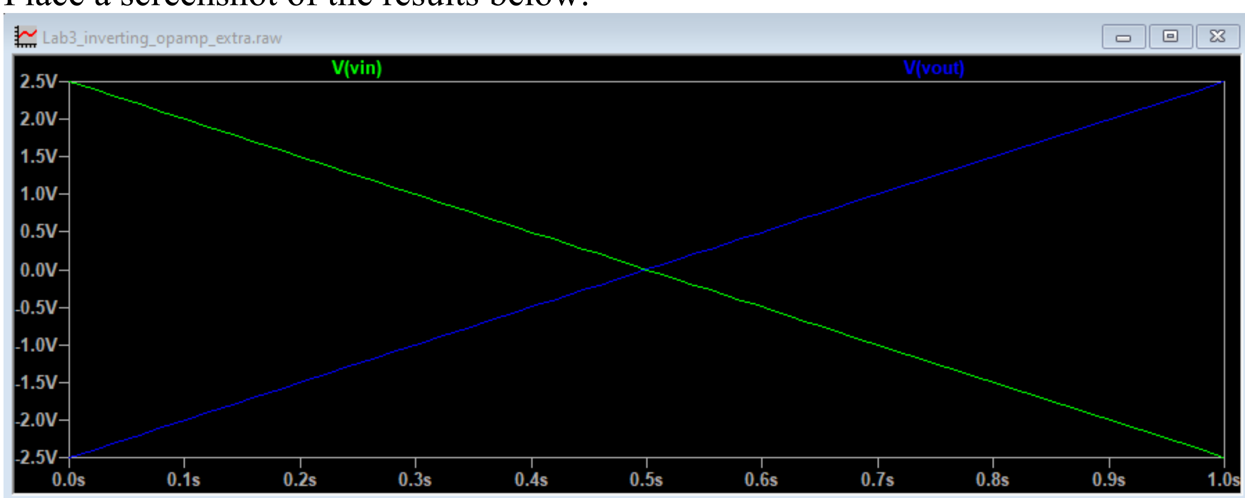
New Commands:

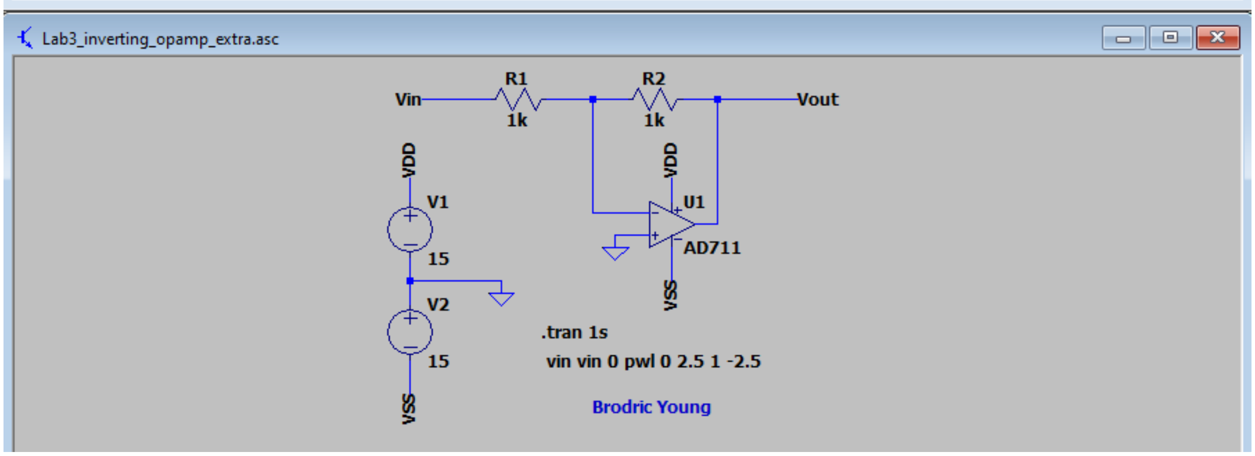
- .tran 1s
  Instructs SPICE to perform a transient simulation for 1 second
- vin vin 0 pwl 0 2.5 1 -2.5
  Instructs SPICE to create a Piece-Wise-Linear voltage source.
    - The name of the voltage source is "vin"
    - The voltage source is connected to two nets (wires) called "vin" and "0". The "0" represents the ground or reference point.
    - At time = 0, the voltage is 2.5V
    - At time = 1 second, the voltage is -2.5V

Simulate the circuit and plot the "vin" and "vout" signals.

To view the results, right-click in the simulation window and select "add traces"

Place a screenshot of the results below:





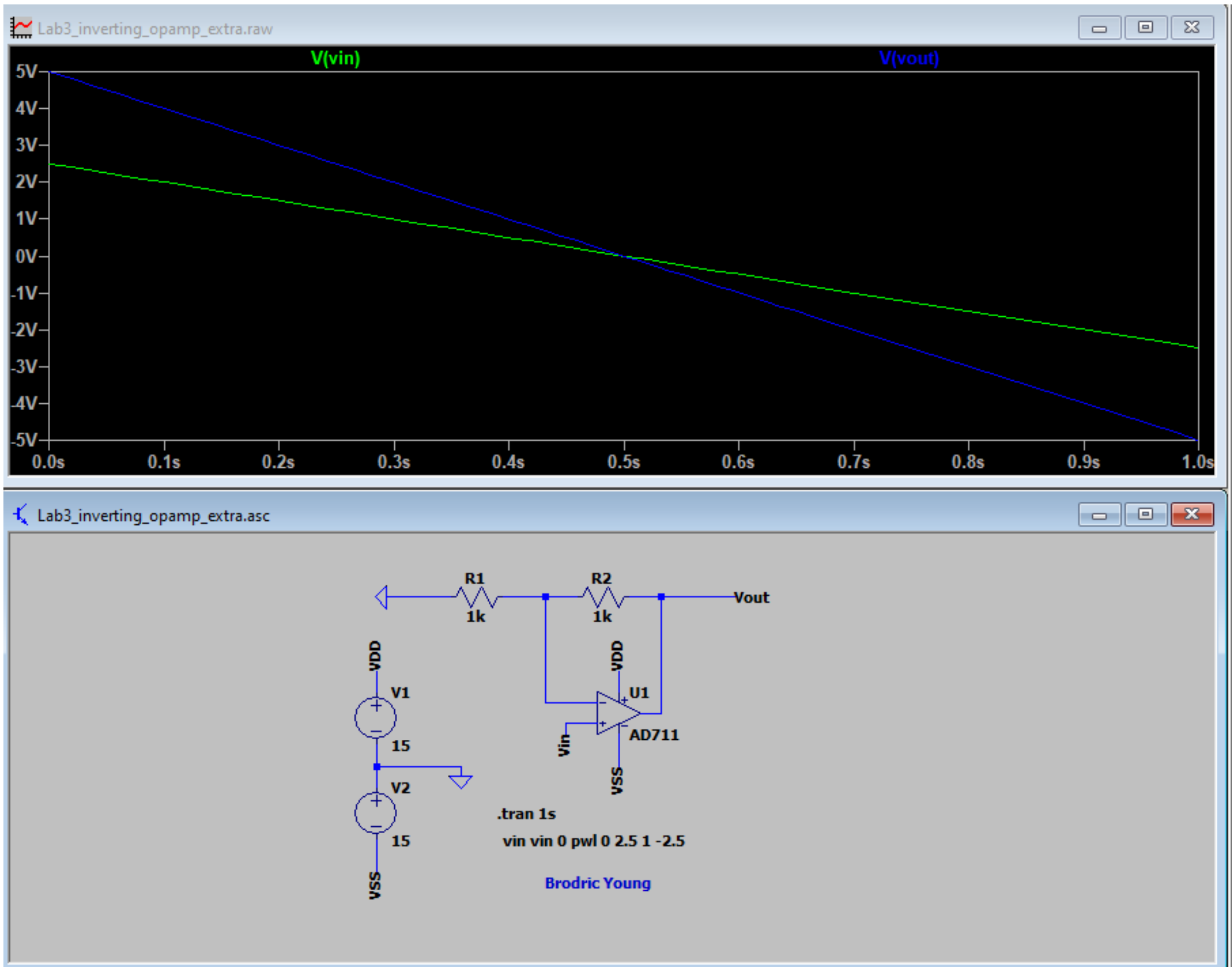
Part 1b - SPICE simulation of a non-inverting amplifier

Modify the schematic to simulate an amplifier with a non-inverting gain of 2. Place a screenshot of your schematic below:

Simulate the circuit and plot the "vin" and "vout" signals.

> To view the results, right-click in the simulation window and select "add traces"

Place a screenshot of the results below:



Reminder of LTspice function keys and shortcuts:

- Esc to exit
- F2 to place a component

    - "r" for resistor
    - "vo" for voltage source
    - "cu" for current source
    - "e" for VDVS
- F3 to place a wire
- F4 to place a net name (like N1, a, and b)
- F4 to place a ground symbol
- F5 to delete
- F6 to copy
- F7 to move
- F8 to drag
- Ctrl-r to rotate
- Ctrl-e to mirror
- Right-click on text to change
- Right-click -> Draft -> SPICE directive to place a command

Part 2a - Differential Amplifier Simulation

Download and simulate the differential amplifier provided with the lab instructions. For this simulation, you will also need the "LF347_model.txt" file. Make sure it is located in the same folder as your schematic file before performing the simulation.

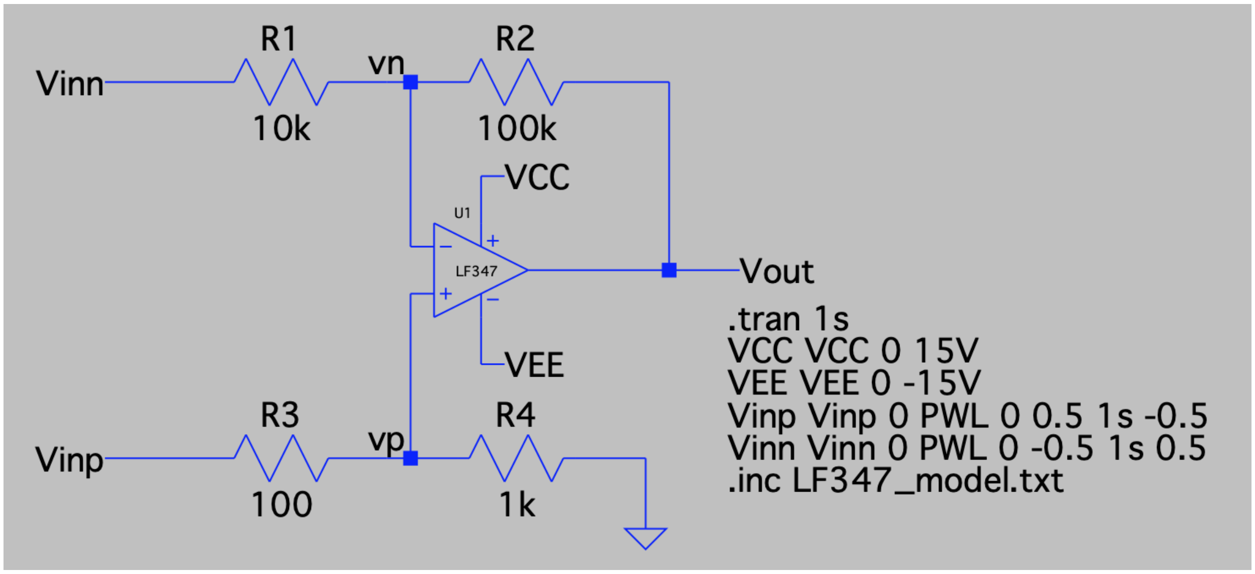
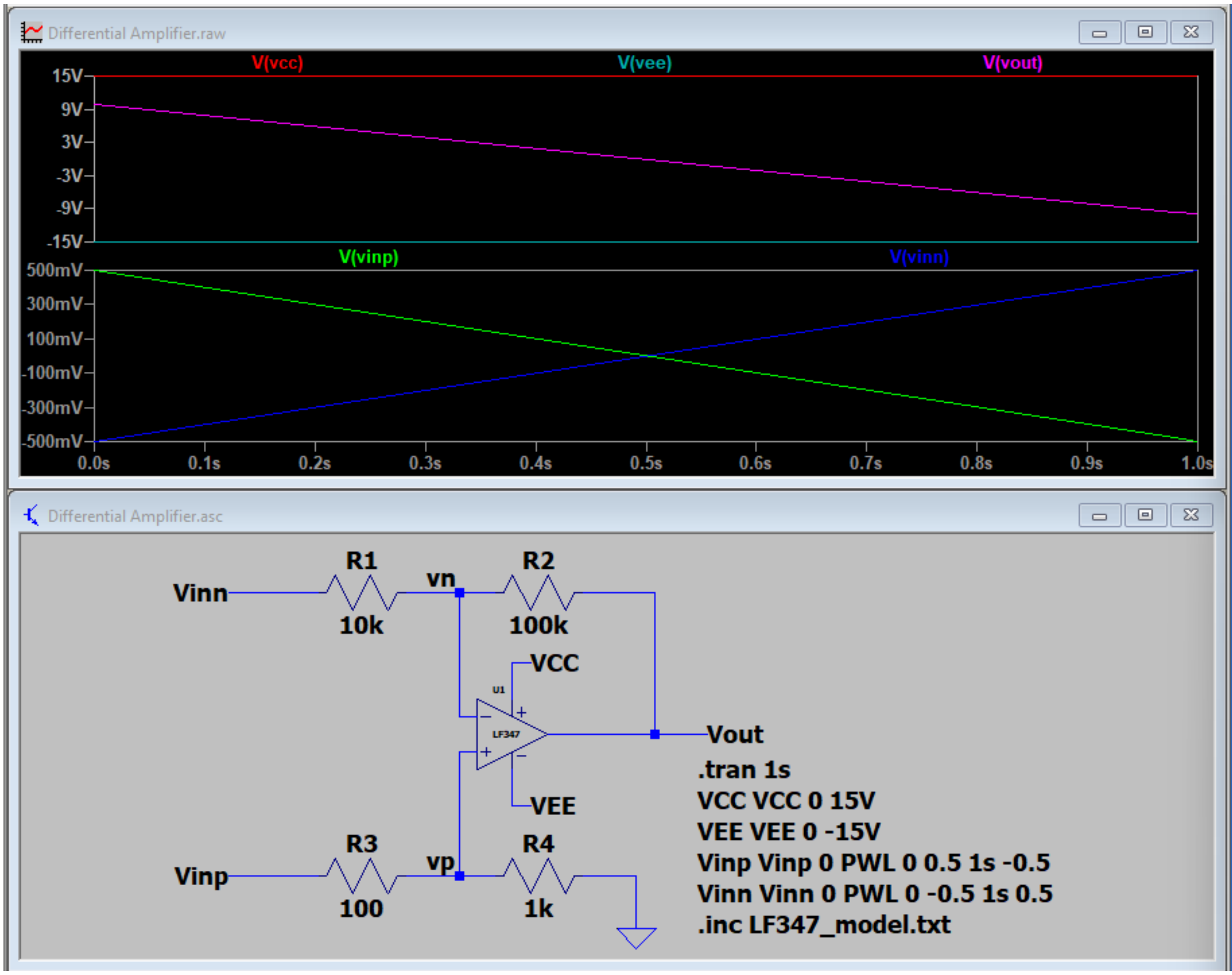


Figure 2 – Schematic of the differential amplifier simulation file

Include a screenshot of the following simulated waveforms in two separate plot panes:

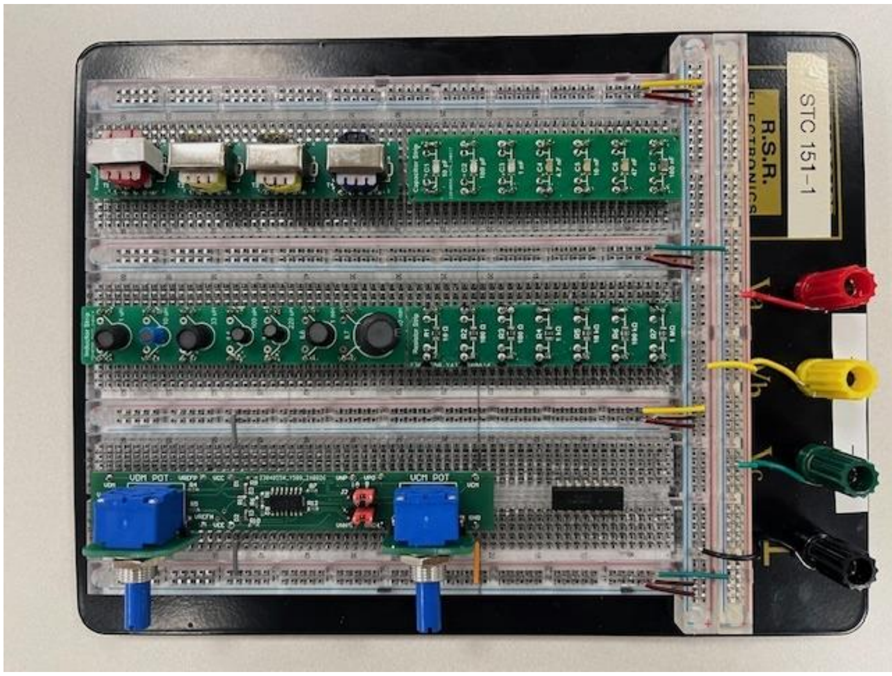
- In plot pane 1, Vinp and Vinn
- In plot pane 2, VCC, VEE, and Vout.

Note: To add a plot pane, right-click on the plot window.

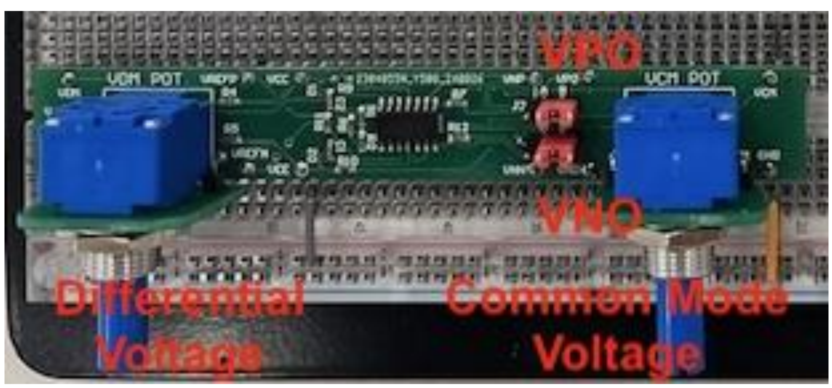

Differential Amplifier.raw
V(vcc)
V(vee)
V(vout)
15V
9V
3V
-3V
-9V
-15V
V(vinp)
V(vinn)
500mV
300mV
100mV
-100mV
-300mV
-500mV
0.0s
0.1s
0.2s
0.3s
0.4s
0.5s
0.6s
0.7s
0.8s
0.9s
1.0s
Differential Amplifier.asc
R1
R2
Vinn
vn
10k
100k
VCC
U1
LF347
Vout
.tran 1s
VCC VCC 0 15V
VEE
VEE VEE 0 -15V
R3
R4
Vinp
vp
Vinp Vinp 0 PWL 0 0.5 1s -0.5
100
1k
Vinn Vinn 0 PWL 0 -0.5 1s 0.5
.inc LF347_model.txt

Part 2b - Differential Amplifier Construction

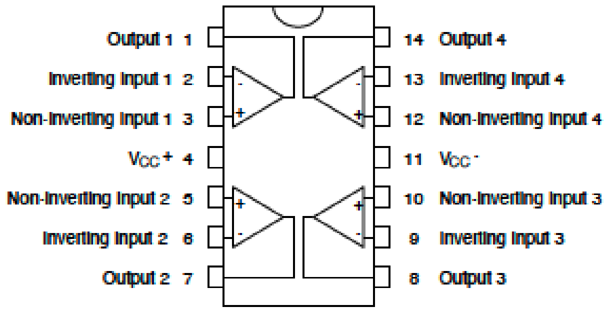
Construct the differential amplifier circuit you simulated using the ECEN250 breadboard system and make the specified measurements.



ECEN250 Breadboard system



Differential Voltage Reference Board



Pin Connections of LF347

Measure Vout under the input conditions listed in the table. If you can't produce the exact input voltages, replace the target values with the measured values:

Target Vinn (V)	Target Vinp (V)	Vout
-0.5	0.5	9.2 V
0.5	-0.5	-10.6 V
4	4	-0.8 V
-4	-4	0.17 V

Calculate the differential gain:

Ideal Example: $A_{DM} = \Delta Vout/\Delta Vin_{Diff} = 10V - (-10V) / (1V - (-1V)) = 10V/V$

$A_{DM} = 9.9$

Calculate the common mode gain:

Ideal Example: $A_{CM} = \Delta Vout/\Delta Vin_{CM} = 0V - (-0V) / (4V - (-4V)) = 0V/V$

$A_{CM} = 0.12$

Calculate the CMRR:

CMRR = 82.5

Conclusions (write a conclusion statement that discusses each of the purposes of the lab):

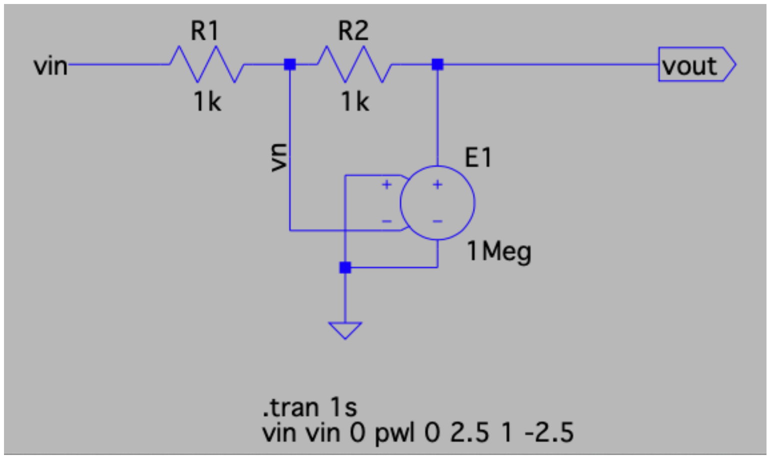
In this lab, we simulated both an inverting and non-inverting gain configuration of an op amp using LTspice. Switching from one to the other really emphasized how similar they are, you just switch which line the power comes in at and which is the ground. The graphs shown in the simulation were very helpful to see the behavior of op amps. We also simulated a differential amplifier to understand what is was doing exactly. Also to gain more understand we constructed this differential amplifier on the breadboard using the two potentiometers to adjust to differential mode and common modes. We made voltage measurements for the output after setting the input to certain values to show that this op amp amplified a difference in input voltages and that when there wasn't a difference, it would result in 0 for the output. We also calculated the DM gain and CM gain which we as we expected with a 10:1 ratio in the resistors.

10% Extra Credit Opportunity #1:
Apply a 1kHz, 2Vp-p sine wave to the input using a function generator. Insert an oscilloscope image of the waveform:

10% Extra Credit Opportunity #2

SPICE simulation of an inverting amplifier

Simulate the following circuit in LTspice:



A SPICE Model of an Operational Amplifier Using a VDVS

Note: to insert a VDVS, use the "F2" function key and enter the letter "e" into the search filter.

Description of the circuit:

This circuit uses a Voltage Dependent Voltage Source to emulate the function of an operational amplifier. The open loop gain of the amplifier is 1 million. Note that the center-tap of the two resistors is connected to the negative input and the positive input is connected to ground.

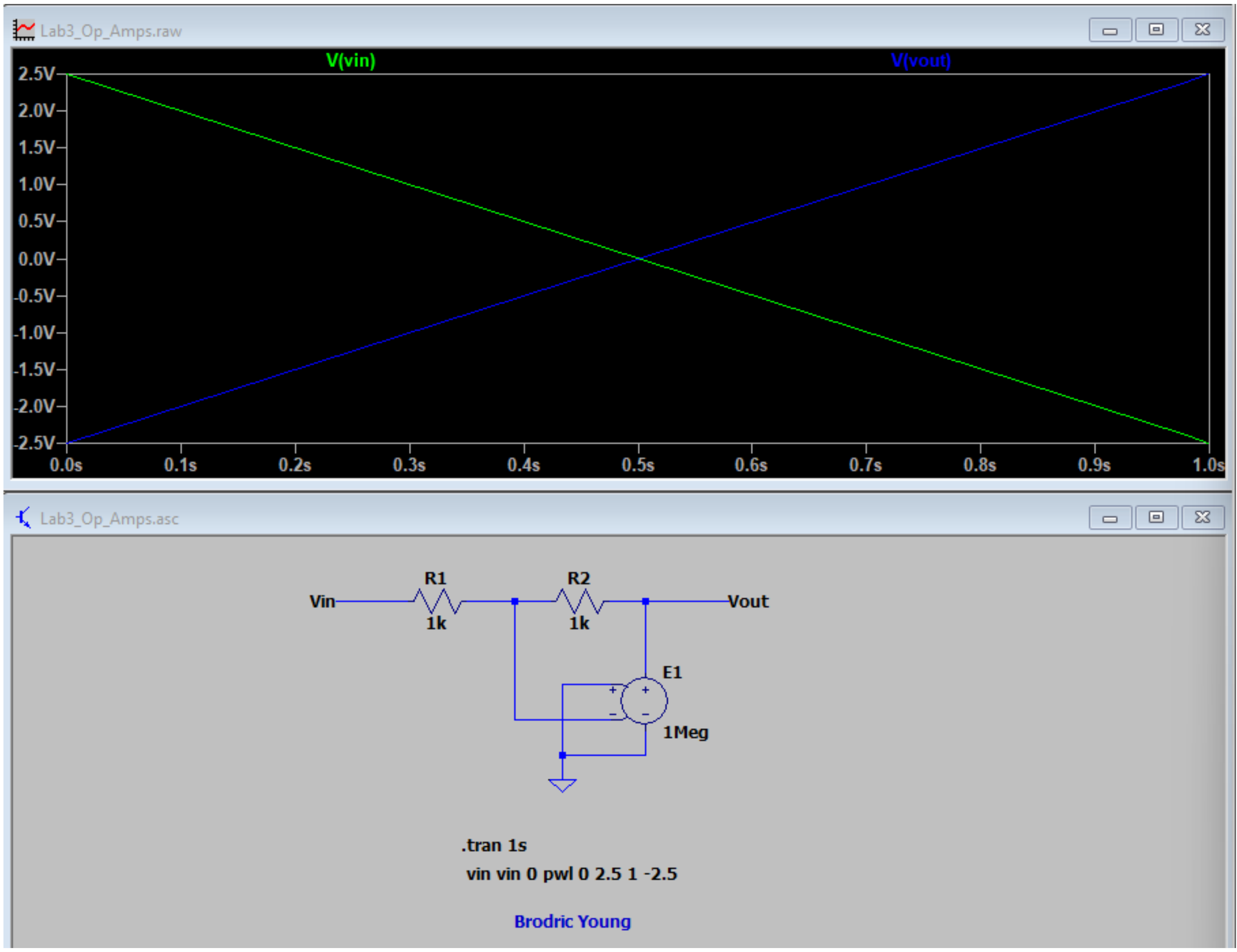
New Commands:

- .tran 1s
  Instructs SPICE to perform a transient simulation for 1 second
- vin vin 0 pwl 0 2.5 1 -2.5
  Instructs SPICE to create a Piece-Wise-Linear voltage source.
  - The name of the voltage source is "vin"
  - The voltage source is connected to two nets (wires) called "vin" and "0". The "0" represents the ground or reference point.
  - At time = 0, the voltage is 2.5V
  - At time = 1 second, the voltage is -2.5V

Simulate the circuit and plot the "vin" and "vout" signals.

To view the results, right-click in the simulation window and select "add traces"

Place a screenshot of the results below:



SPICE simulation of a non-inverting amplifier

Modify the schematic to simulate an amplifier with a non-inverting gain of 2. Keep using the VDVS to represent the op amp in the circuit. Place a screenshot of your schematic below:

Using the same simulation parameters, simulate the gain of the non-inverting amplifier with an A_{OL} of 1MEG. Record your simulation results in the table below. Change the A_{OL} of the VDVS component to the values below and record the simulated results:

A_{OL}	ΔVout	ΔVin	Simulated A_{CL}	Calculated A_{CL}
1MEG				
10k				
100				
1				

The A_{CL} equation for a non-inverting amplifier configuration is:

$$A_{CL} = \frac{A_{OL}}{1 + \beta A_{OL}}$$

Where β is the feedback factor and is derived from the voltage divider equation:

$$\beta = \frac{R_2}{R_2 + R_1}$$

With R_2 being the feedback resistor.

Does the equation for A_{CL} match the simulated values?